

Available online at [www.sciencedirect.com](http://www.sciencedirect.com)**ScienceDirect**

Procedia Engineering 135 (2016) 316 – 321

**Procedia  
Engineering**[www.elsevier.com/locate/procedia](http://www.elsevier.com/locate/procedia)

# Numerical simulations of LNG vapor dispersion from LNG jetting in different directions

Qian-xi Zhang<sup>a,b,c</sup>, Dong Liang<sup>b,c,\*</sup><sup>a</sup>*School of Engineering, Guangdong Ocean University, Zhanjiang 524008, China*<sup>b</sup>*School of Engineering, Sun Yat-sen University, Guangzhou 510006, China*<sup>c</sup>*Guangdong Provincial Key Laboratory of Fire Science and Technology, Guangzhou 510006, China*

## Abstract

LNG jetting release in emergency pressure relief or storage tank leakage accident is common. Computational fluid dynamics model of LNG vapor dispersion from LNG jetting is established, and we performed numerical simulations in a more general situation of three jet directions, then we got the result of CH<sub>4</sub> concentration distribution in jet diffusion area. Depending on the results of the comparison, we found that LNG vapor diffusion distance is the farthest when jetting along the direction of the wind. The results provide technical support for LNG storage unit emergency relief pipe installation choices and jet leakage accident emergency management.

© 2016 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license

(<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of the organizing committee of ICPFFPE 2015

**Keywords:** LNG, jetting release, CFD

## 1. Introduction

Natural gas is used more and more as a kind of high quality clean energy. Day by day serious atmospheric pollution makes China speed up the development of clean energy. Natural gas real environmental and social value has received increasing attention. China builds multiple LNG receiving station, LNG imports in 2013 increased by more than 20% than in 2012. China's natural gas will account for 7.5% of the total energy consumption in 2015. Liquefied natural gas (LNG) is a key technology of the development and utilization of natural gas at home and abroad. LNG has formed a new industry and continues to grow at an average rate of 8% [1].

At the same time, the safe storage and use of LNG has caused people's great attention. When the pressure in the LNG storage device exceeds the permissible value, emergency relief pipe will be open, and LNG vapor will spray from device. In addition, the damaged LNG vessel or pipe also can bring LNG leakage jet. Once the LNG leakage occurs, LNG vapor can cause frostbite in low temperature, suffocation. Fire and explosion easily happen when the vapor meets the direct fire.

Consequence modeling of accidental LNG releases has been studied extensively as part of the effort extended to prevent and mitigate such incidents. At first, integral models, such as DEGADIS, SLAB, HEGADAS, and many others, are widely used because of their fast computational time and ease of use [2-4]. However, most integral models have limitations in describing the terrain and congestion density of obstacles in LNG spill scenarios, whose effects on vapor cloud and turbulence cannot be neglected in modeling vapor dispersion [5]. Afterwards, advancements in computation capabilities, including processing capacity and memory space, have made it possible for engineers to use computational fluid dynamics

\* Corresponding author. Tel.: +86-20-39332230;

E-mail address: 14083726@qq.com

(CFD) models to solve complex fluid flow problems. CFD models are able to provide a detailed description of physical processes and handle complex geometries, and can thus be used to predict the behavior of LNG vapor cloud dispersion in a site-specific risk analysis [6–11]. However, CFD simulation setup methods for LNG jet release and vapor dispersion and have not been sufficiently reported in the literature.

In the present work, ANSYS fluent 11.0 was used to perform simulations of LNG jetting in three direction and vapor dispersion. Thus, a sensitivity analysis was conducted to illustrate the impact of the mesh size and source term turbulence intensity on predicting distance to the lower flammable limit (LFL). The motivation of this work was to provide guidance in modeling LNG jetting and LNG vapor dispersion with ANSYS fluent, which can be used to emergency relief pipe installation choices.

## 2. Domain and mesh

Wrong operation or man-made damage all can bring about liquefied natural gas injection leaks. This research object is LNG jetting in different directions and the calculation domain which is large enough for the simulation. The simulation domain as shown in the figure1 as following is selected., and X, Y, Z, respectively (mm): 25000, 7000, 10000, and the X direction parallel to the wind direction. The second step is subdividing the computational domain. The domain is discretized into a number of small control volumes using a mesh. The mesh size must be chosen carefully to avoid the adverse effect on the simulation accuracy. The grid was finest close to the ground and the injection orifice. A recommended approach to eliminate mesh size influence is to seek mesh-independent solutions by testing with gradually reduced mesh sizes until the simulation results no longer change. In this case, the domain was discretized into about 1349000 cells, as shown in fig.1, which was enough to provide grid independent results.

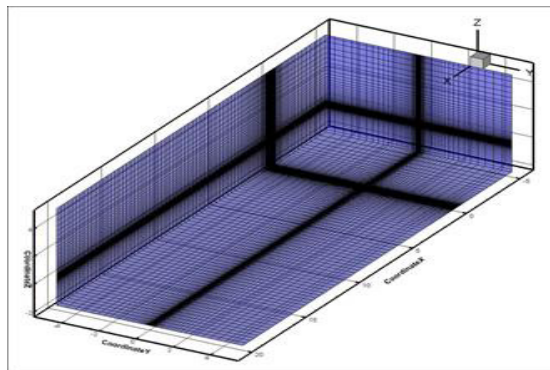


Figure1. Domain and mesh

## 3. Model

A turbulence model must be identified to predict the effects of turbulence in the ambient atmosphere and natural gas. ANSYS FLUENT 11.0 offers a large variety of turbulence models, such as the  $k-\epsilon$  model,  $k-\omega$  model, and shear stress transport (SST) model [12]. A comparative study of these turbulence models against experimental data has been reported elsewhere [13]. In the present work, the standard  $k-\epsilon$  model was used because of its balance between computational time and precision. This model has been used for numerical simulations of LNG vapor dispersion and other gas dispersions with satisfactory results [14,15,16].

At the same time, heat transfer model and species transport model are selected. Heat transfer model is to represent the heat transfer throughout fluids within the domain. This model must take into account both the thermal energy and kinetic energy, which can be addressed in ANSYS FLUENT 11.0 using the total energy model.

### 3.1. Viscous Model

Realizable  $k-\epsilon$  model is a turbulence model developed in the modern times. It differs from the standard  $k-\epsilon$  model in the two aspects below: It provides a new analytic expression for calculating turbulence viscosity; for turbulence dissipation rating calculation, it provides a new transportation formula obtained based on the accurate transportation formula

describing mean square vortex fluctuation. The Realizable  $k - \varepsilon$  turbulence flow transportation model is highly reliable and accurate and has improved the  $\varepsilon$  equation. The rotary boundary-layer flow and the complicated secondary flow apply to all the flow forms in  $k - \varepsilon$  and RNG  $k - \varepsilon$ . On this basis, ANSYS Fluent11.0 software and the Realizable  $k - \varepsilon$  substance transportation and reaction module were used for the simulation analysis of the dispersion of spilled LNG. This model was used for the simulation solution of transportation of spilled LNG.

In the Realizable  $k - \varepsilon$  model, the turbulent kinetic energy transportation equations about  $k$  and  $\varepsilon$  are as below: [9]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (1)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) &= \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \\ &+ \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \end{aligned} \quad (2)$$

In the equations,  $C_1$ ,  $C_2$ ,  $C_{1\varepsilon}$ ,  $C_{3\varepsilon}$ ,  $\sigma_k$  and  $\sigma_\varepsilon$  are all transportation dissipation constants;  $u_j$  and  $x_j$  are the transportation speed and distance in direction  $j$ ;  $G_k$  refers to the turbulent kinetic energy source term generated because of the uniform velocity gradient;  $G_b$  refers to the turbulent kinetic energy source term generated because of the buoyancy;  $Y_M$  refers to the dissipation term generated because of fluctuation and expansion in a compressible turbulent flow;  $S_k$  and  $S_\varepsilon$  are customized conditions;  $\rho$  is subject to the compressible gas state equation.

### 3.2. Species Model

By solving the substance transportation dispersion conservation equation, Fluent can predict the mass fraction of substance  $i$ . The substance transportation dispersion equation is as below: [8]

$$\frac{\partial}{\partial t}(\rho \omega_i) + \nabla \cdot (\rho \omega_i \vec{v}) + \nabla \cdot \vec{J}_i = R_i + S_i \quad (3)$$

In the equation,  $\vec{v}$  is the transportation speed vector;  $R_i$  is the speed of substance  $i$  generated in the chemical reaction (if any), or may be neglected when there is no chemical reaction;  $S_i$  is the speed of substance  $i$  generated based on customization;  $\vec{J}_i$  is the mass diffusion speed of substance  $i$ .

## 4. Results and discussion

The analysis presented in this paper represents a generic solution method for simulation of vapor dispersion from natural gas jets into the ambient atmosphere in various jet orientations under the same boundary condition.

### 4.1. Natural gas dispersion under different jet directions

5. Wind speed is a common wind speed for the Pearl River Delta region ( $v_a=2.3\text{m/s}$ ,  $v_a$  is the wind speed), along X axis. Pinhole and release rate is constant ( $v_n=2\text{kg/s}$ ,  $v_n$  is the natural gas release rate). The paper compares the effects of three ways on jet direction: along wind, against the wind and perpendicular to the wind direction.

6. After a steady-state wind field is obtained, it was taken as the initial condition, a LNG jet source was opened. After simulating, vertical contour slices of natural gas mole fraction from the lower limit of explosion (5%) of CH<sub>4</sub> and the upper limit of explosion (15%) along the direction of the wind are shown in figures 2-4.

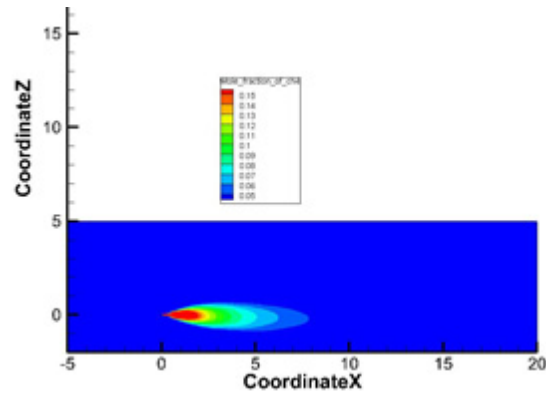


Fig2. Horizontal direction (Jet along the wind)

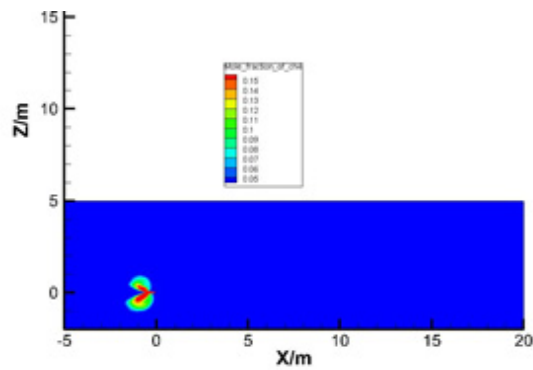


Fig3. Horizontal direction (Jet against wind direction)

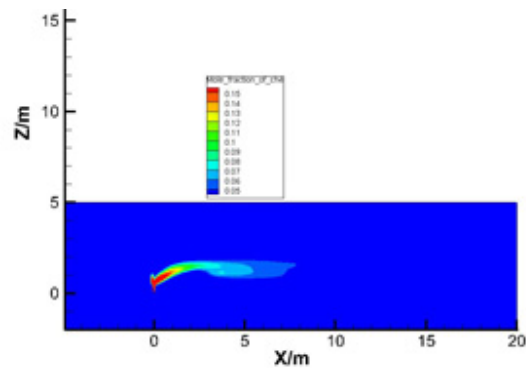


Fig4. Vertical direction

We can see from figures 2-4, the vertical diffusion distances are very alike, while horizontal diffusion distances are different. At the same time, we also see that the danger zone mainly located in the downwind of injection point.

### 1. Result analysis

According to the simulation results above, the concentration field distributions can be gotten. Usually, after the LNG injection spill, we should pay special attention to the diffusion field which concentration is greater than the lower explosive limit of methane. In the computation, monitoring points were set in the center line along the wind direction at horizontal height of 2 meters, respectively. According to the result of comparison of concentration changes of the monitoring points (Fig.5), it is clear that the CH<sub>4</sub> concentration monitoring points of the horizontal direction jet along the wind are larger than that of the monitoring points than other jet ways. This result shows that horizontally jet along the wind direction means more danger.

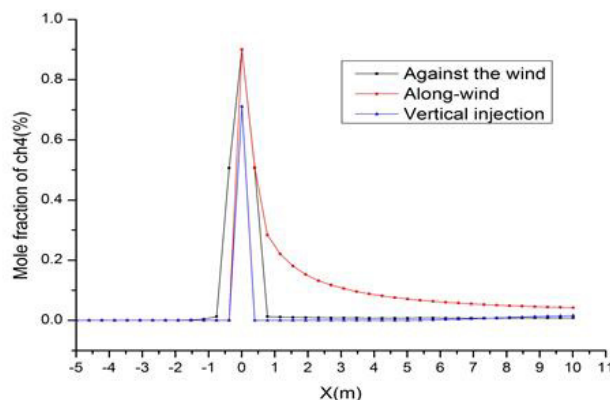


Fig.5 CH<sub>4</sub> concentration on the center horizontal line

### 2. Conclusion

The results show that horizontally jet along the wind direction lead to CH<sub>4</sub> vapor will were able to diffuse farther and broader in scope along the horizontal direction. It also shows that if the LNG container needs to install the emergency pressure relief pipe, pipe exports should not adopt a horizontal direction.

### Acknowledgements

This work was supported by grants from the Open-end Fund of Guangdong Provincial Key Laboratory of Fire Science and Technology, the Guangdong Provincial Scientific and Technological Project (No. 2011B090400518), the Guangdong Provincial Key Laboratory of Fire Science and Technology (No. 2014B030301034), the Zhanjiang Science and technology research projects, the experimental teaching reformation project of the National Information Technology Experimental Teaching Demonstration Center of Sun Yat-sen University, 2103.

### References

- [1] Gu Anzhong, Lu Xuesheng. Liquefied natural gas (LNG) technical manuals. Beijing: China Machine PRES, 2014:21-27.
- [2] T.O. Spicer, J.A. Havens, Field test validation of the DEGADIS model, J. Hazard. Mater. 16 (1987) 231–245.
- [3] D.L. Ermak, S.T. Chan, D.L. Morgan, L.K. Morris, A comparison of dense gas dispersion model simulations with burro series LNG spill test results, J. Hazard. Mater. 6 (1982) 129–160.
- [4] J.S. Puttock, Comparison of Thorney Island data with predictions of HEGABOX/HEGADAS, J. Hazard. Mater. 16 (1987) 439–455.
- [5] A. Luketa-Hanlin, A review of large-scale LNG spills: experiments and modeling, J. Hazard. Mater. 132 (2006) 119–140.
- [6] A. Luketa-Hanlin, R.P. Koopman, D.L. Ermak, On the application of computational fluid dynamics codes for liquefied natural gas dispersion, J. Hazard. Mater. 140 (2007) 504–517.
- [7] S.T. Chan, Numerical simulations of LNG vapor dispersion from a fenced storage area, J. Hazard. Mater. 30 (1992) 195–224.
- [8] F. Gavelli, E. Bullister, H. Kytomaa, Application of CFD (fluent) to LNG spills into geometrically complex environments, J. Hazard. Mater. 159 (2008) 158–168.
- [9] S. Sklavounos, F. Rigas, Simulation of Coyote series trials—Part I: CFD estimation of non-isothermal LNG releases and comparison with box-model predictions, Chem. Eng. Sci. 61 (2006) 1434–1443.

- [10] O.R. Hansen, J.A. Melheim, I.E. Størvik, CFD modeling of LNG dispersion experiments, in: AIChE Spring National Meeting, Houston, TX, 2007.
- [11] B.R. Cormier, R. Qi, G. Yun, Y. Zhang, M.S. Mannan, Application of computational fluid dynamics for LNG vapor dispersion modeling: a study of key parameters, *J. Loss Prevent. Proc. Ind.* 22 (2009) 332–352.
- [12] ANSYS FLUENT 11.0 Tutorial Manual
- [13] S. Sklavounos, F. Rigas, Validation of turbulence models in heavy gas dispersion over obstacles, *J. Hazard. Mater.* 108 (2004) 9–20.
- [14] G.A. Perdikaris, F. Mayinger, Numerical simulation of heavy gas cloud dispersion within topographically complex terrain, *J. Loss Prevent. Proc. Ind.* 7 (1994) 391–396.
- [15] F. Gavelli, E. Bullister, H. Kytömaa, Application of CFD (fluent) to LNG spills into geometrically complex environments, *J. Hazard. Mater.* 159 (2008) 158–168.
- [16] S. Sklavounos, F. Rigas, Simulation of Coyote series trials—Part I: CFD estimation of non-isothermal LNG releases and comparison with box-model predictions, *Chem. Eng. Sci.* 61 (2006) 1434–1443.